ISSUE 1 | 2020

ReSolve

Productivity - Reliability - Usability

COVID-19 Pandemic

Revolutionizing Design Through Advanced Simulation

PAGE 26

AeroDelft

A sustainable future with liquid hydrogen aircraft

PAGE 32

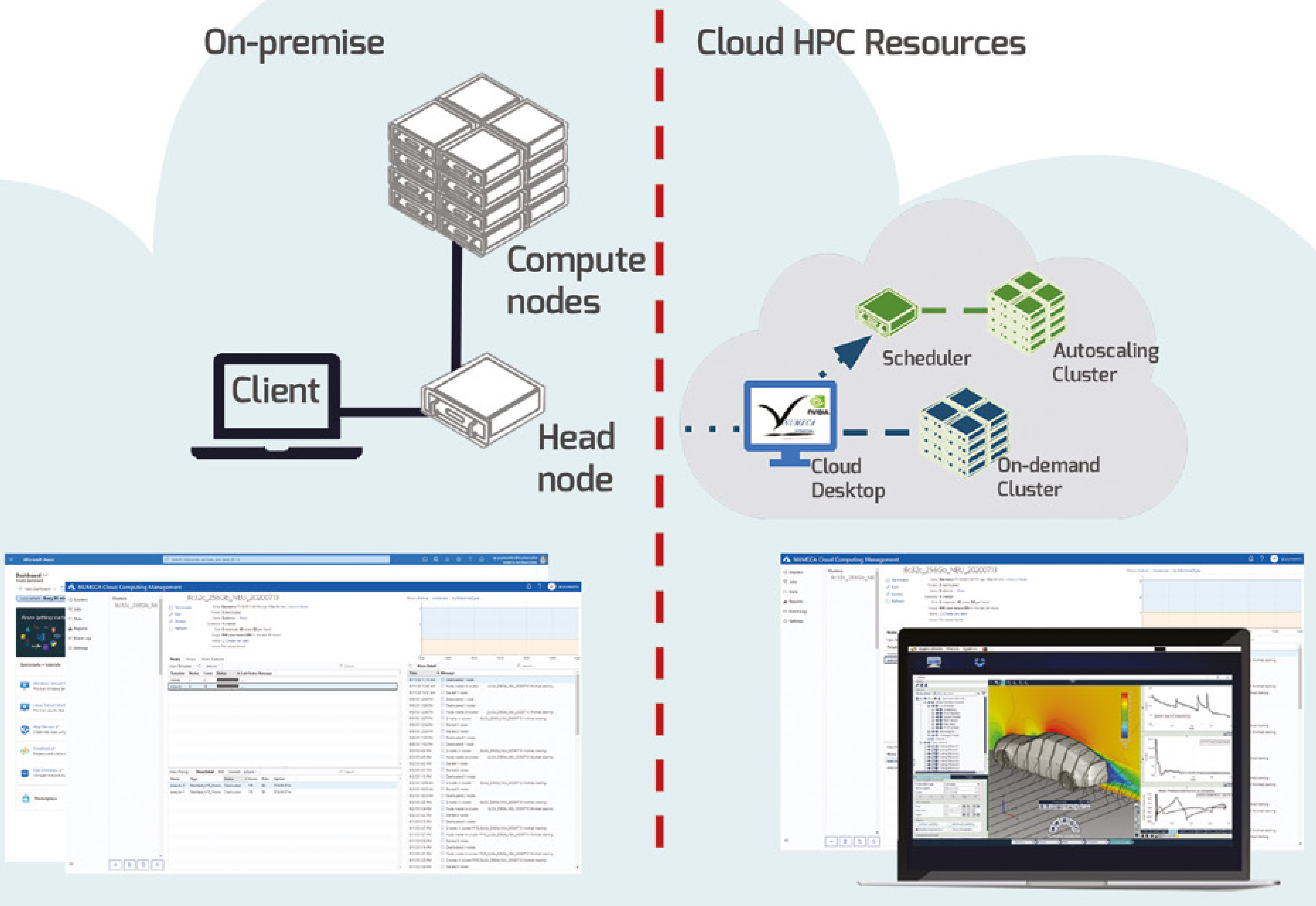
Teknicraft

A new era of sustainable ecotourism through design optimization of whale watching vessels

PAGE 36

NUMECA INTERNATIONAL

Be Productive
Be Flexible
Be Fast



IT Team

Engineering Team

Unlock your innovation with NUMECACloudPower:

Access any NUMECA software via your web browser
with as many HPC resources as you need!

NUMECA INTERNATIONAL

Microsoft Azure

Uber Cloud

The Next Generation of CFD Technologies

This Third edition of our ReSolve Newsletter, covering diverse application areas of analysis and design optimization, brings a testimony of NUMECA's steps along the road towards the next generation of CFD technologies.

Until recently, CFD vendors were essentially focused on enhancing performance of their solvers, an area where NUMECA has demonstrated to be at the forefront of the competition with its outstanding levels of convergence speed.

But the current global awareness of sustainability in all areas of human and industrial activities has driven the focus towards a wider scope of essential functionalities characterized by enhanced engineering productivity of the complete chain of simulation. This impacts all levels of the industrial design process of innovative and competitive products, based on virtual prototyping. In these challenging times of worldwide pandemic, cost reduction through enhanced productivity has become a major necessity in many industrial areas.

The key components of this new approach, in service to our industrial customers, is expressed by the combination of three major concepts: *Reliability, Speed & Useability*.

Reliability: is the response to the essential question that is guiding the technology developments at NUMECA, namely: how do we minimize the risks associated with the industrial design process based on simulations, taking into account all elements entering the simulation process and their numerous associated uncertainties?

This requires a multi-step approach, starting with the quality of our software tools, followed by a dedicated interaction with our customers to optimize our modeling chain within the innovative OMNIS™ environment, via an application-based focus. This is followed by fine-tuning NUMECA's advanced optimization framework which is extended to a global strategy of Robust Optimization (RO), taking into account the main uncertainties affecting the operations of the industrial product. At this stage, NUMECA has the most advanced RO system on the market, due its long expertise in Uncertainty Quantification.

Speed: Achieving the highest speed levels in the turn-around time of our solvers has required major innovative developments, including our well-known CPUBooster technology, as well as the Nonlinear Harmonic (NLH) unique methodology. In addition, major developments have led to full exploitation of High Performance Computing (HPC), including Cloud capacity, leading to scalabilities up to many tens of thousands of processors, as well as adaptation to GPU's.

Useability: is a major component of the productivity objectives, in support of our users. This is achieved by customizing the whole simulation chain to the specific needs of each individual user, based on the flexibility of the OMNIS™ set-ups and advanced pre-and post-processing functionalities.

The contributions to this ReSolve issue illustrate many facets of the above strategy and of our persistent developments towards a new generation of CFD technologies.

The next issue will be focused on the further steps NUMECA is undertaking in revolutionizing the world of CFD.

Prof. Charles Hirsch,
President, NUMECA International

In this issue

Revolutionizing the design of particle counters

through advanced simulation to study the transmission mechanisms of COVID-19 - Particles Plus

PAGE 26

Aerodynamic optimization of liquid hydrogen aircraft

pushes the airline industry towards a sustainable future - Aerodelft

PAGE 32

Robust Design Optimization

of ship propellers lowers fuel consumption and operational costs - NUMECA International

PAGE 36

Editorial Team: **Joris Vanherzeele**, *Editorial Director*, **Anne-Marie Schelkens** and **Rebecca Watrous**, *Senior Editors*.

Would you like to be featured in next issues?
Contact us: info@numeca.com

Mitigating noise emissions of Distributed Electric Propulsion (DEP) systems for electric aircraft

Pipistrel is a world-leading designer and manufacturer of small aircrafts specializing in energy-efficient and affordable high-performance aircraft. The first company to fly an electric two-seater in 2007 and the winner of the NASA Green Flight Challenge in 2011 (with the world's first electric four-seat airplane), Pipistrel has designed nine different experimental and serially-produced electric aircrafts, including the first type certified electric airplane, the Velis Electro. Pipistrel has also developed several propulsion systems, including batteries, power controllers and electric motors, for small and general aviation class of aircraft for NASA and Siemens, among others. With involvement in various standardization committees and research projects, Pipistrel is helping to enable the future market of hybrid-electric aviation.

By **Dr Jernej Drofelnik**, *Aerodynamics Design Engineer, Pipistrel Vertical Solutions.*

The ARTEM[1] project and Distributed Electric Propulsion

One such research project that Pipistrel is participating in is the Aircraft noise Reduction Technologies and related Environmental iMpact (ARTEM) project. The ARTEM project's objective is to develop novel technologies to reduce aviation noise emissions for the engines and airframes of future aircrafts. Pipistrel is leading the investigation and mitigation of noise emissions of Distributed Electric Propulsion (DEP) systems.

DEP systems, which are gaining popularity in the age of electric aviation, use multiple propulsion units distributed about the airframe. Since they are only connected electrically to energy sources or power-generating devices, the propulsion units can be placed, sized, and operated with greater flexibility and provide improved performance over more traditional designs.[2] Some of the main benefits of this propulsion system are shorter take-off and landing, reduced energy consumption for increased flight ranges, and noise reduction. The technology offers numerous integration capabilities for a wide number of future aircraft concepts.

References

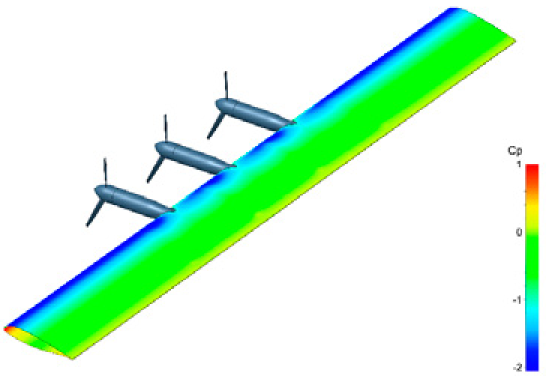
1 ARTEM project received funding from the European Union's Horizon 2020 research and innovation programme under grant No 769 350.

2 A Review of Distributed Electric Propulsion Concepts for Air Vehicle Technology - Kim, Hyun D - NASA Armstrong Flight Research Center, Edwards, California & Perry, Aaron T., and Ansell, Phillip J. - University of Illinois at Urbana, Champaign, Urbana, Illinois.

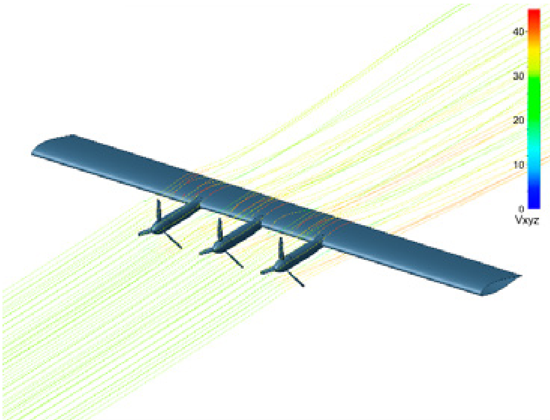
For their role in the ARTEM project, Pipistrel designed and manufactured a DEP mock-up, which serves as a validation platform for various new methods, designs, and low-noise technologies. Its aerodynamic design was based on FINE™/Open solvers using HEXPRESS™/Hybrid's grid generator. For fast aerodynamic design, an actuator disc approximation was used, while for more refined design, steady and unsteady solvers were employed, as shown in Figure 1.

FIGURE 1 : An example of a CFD simulation using MRF approach.

Pressure coefficient plotted on the wing surface



Velocity vectors passing the propellers



Time-Domain Navier Stokes versus Non-Linear Harmonic methods

Pipistrel and its project partners are studying noise source and propagation in the near- and far-field of the DEP design, but the main drawback of using Navier-Stokes CFD for the computation of input data for noise propagation codes (from the designer point-of-view), is its high computational cost. Time-domain (TD) Navier-Stokes simulation of the DEP's periodic flows requires long runtimes, as several rotor revolutions need to be computed before achieving periodic state.

“With the Non-Linear Harmonic (NLH) method implemented in FINE™/Open, these simulation runtimes can be substantially reduced by solving the governing equations in the frequency-domain. This state-of-the-art method can accurately model flow unsteadiness at significantly reduced computational cost and at comparable accuracy to TD method."

Figure 2 shows the evolution of the thrust of one propeller blade over one revolution by both TD and NLH methods. The periodic unsteady perturbations linked to the interaction between the propeller blades and the wing are modeled using Fourier harmonics, whose frequencies are associated to the periodicity and to the relative rotational speed between these two components. From a theoretical viewpoint, an infinite number of harmonics would lead to a perfect match with the traditional time-domain solution. From an engineering viewpoint, the NLH method has proven to satisfactorily capture the same complex unsteady flow phenomena as the TD method, using as few as three complex harmonics.

Figures 3 and 4 compare the pressure distribution on the propeller blade surface for the TD and NLH methods when the blade is aligned with the wing leading edge. For pressure distribution only negligible differences can be noticed on both sides of the blade.

FIGURE 2 : Evolution of thrust for single blade over one revolution.

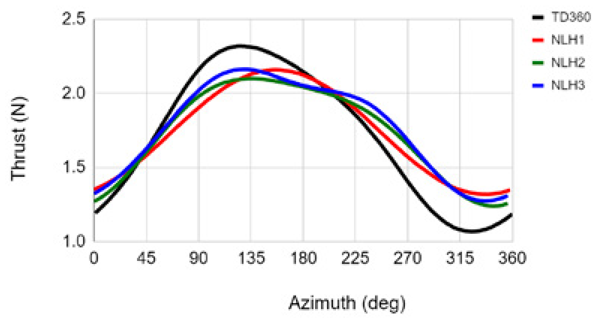
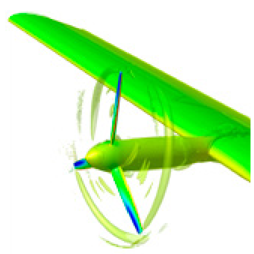


FIGURE 3 : Static pressure coefficient distribution on the wing, propeller and iso surfaces of lambda2 vortex detection criterion.

NLH-3



TD-360

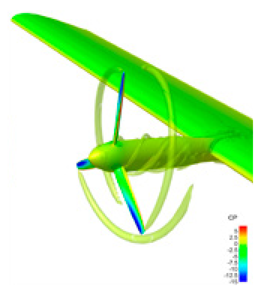
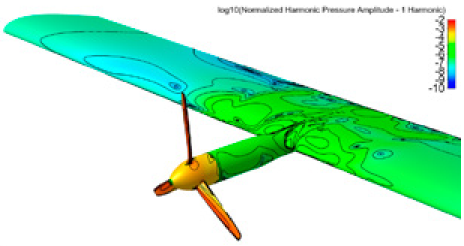
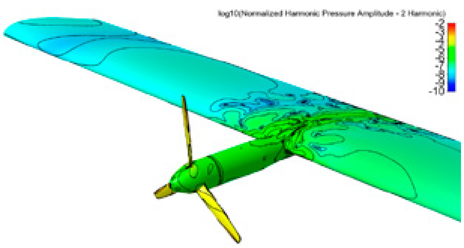


FIGURE 4 : Logarithmic normalized harmonic pressure amplitude distribution over the wing and propeller surfaces (NLH-3 computation).

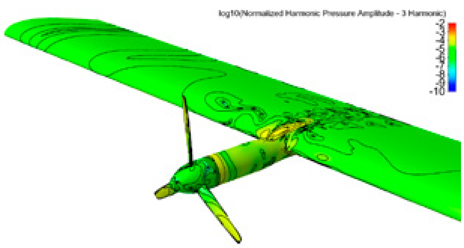
1 Harmonic



2 Harmonic



3 Harmonic



Conclusions

FINE™/Open's NLH method proved to be the most cost effective approach for acoustic optimization analysis of the DEP set-up. The method offered a speed-up between one to two orders of magnitude in comparison to the TD analysis, while providing very similar accuracy.

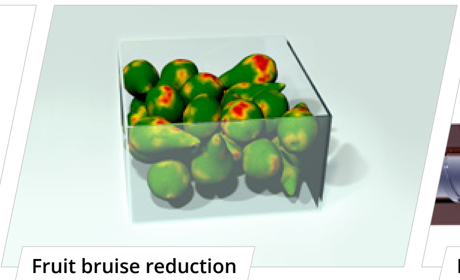
NLH technology offers a fast alternative to conventional TD methods for solving unsteady periodic flows and can easily fit into an industrial workflow.

Ribbon mixer

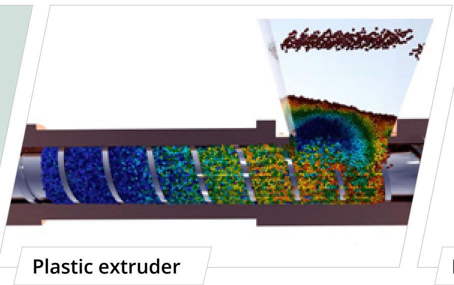
By **Laura Trappolini**, *Head of the Multiphysics Application Group, NUMECA International.*

Simulating the dynamics of particles with Discrete Element Method (DEM) solver OMNIS™/Mpacts

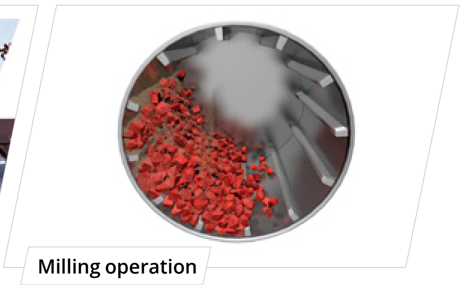
Particles are all around us. Around 80% of all industrial processes are estimated to deal with particulate materials and particle interactions. Examples can be found in food processing (fruit bruise analysis, mixing and blending...), heavy equipment (conveyor belts, hoppers, dump truck...), agriculture equipment (combine harvesters, tractors, tillage...), pharmaceutical and biotech (tablet coating, powder spreading, fluidized bed...) and many more sectors.



Fruit bruise reduction



Plastic extruder



Milling operation

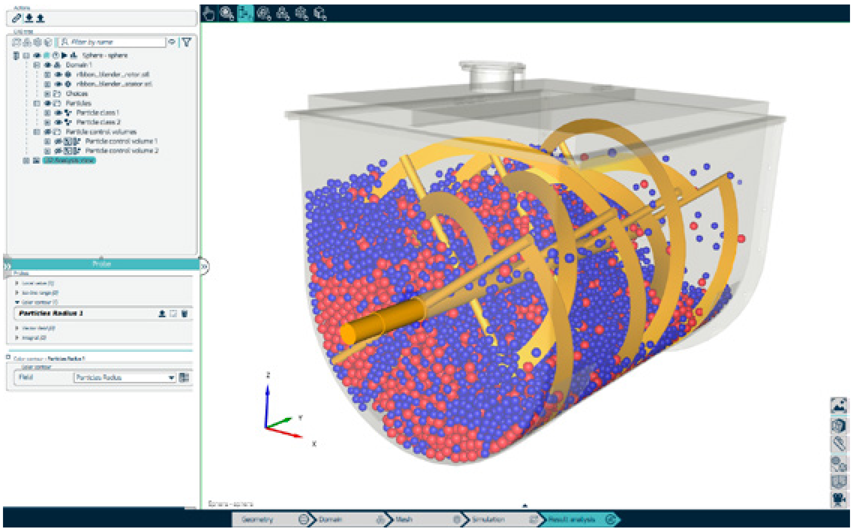
One particular sector in particle management is mixing applications. Some of its main challenges are how to make sure the mixing of all components is homogeneous, and how to ensure this mixing occurs in minimal time.

A concrete example of mixing applications are ribbon mixers. A ribbon mixer is composed of a helicoidal rotor that rotates inside the mixer and a stator called the container. Ribbon mixers are mainly used in the chemical, pharmaceutical, and food industries; their main purpose is to provide efficient mixing of different-shaped particles as quickly and as uniformly as possible.

To ensure effective mixing, companies used to build plenty of expensive prototypes, corresponding to different ribbon mixer designs, in order to experimentally test the mixing process for their particular particle mixes. Today, new software technologies have the potential to help designers of industrial processes understand, analyze, and solve these complex system properties without the need for experimental prototypes. Because of the discontinuous nature of granular materials, a different solver approach compared to the classical Computational Fluid Dynamics (CFD) is required for these applications.

Originally developed at the University of Leuven, in Belgium, Mpacts is a simulation software that uses the Discrete Element Method (DEM): this method simulates the dynamics of large numbers of interacting particles.

FIGURE 1 : DEM simulation of a ribbon mixer with OMNIS™/Mpacts.



The main strength of Mpacts is its huge flexibility in representing the shapes and interaction mechanics of the particles, leading to highly accurate and reliable simulations. Compared to the Finite Element Method, the fundamental difference (and advantage) is that DEM does not need a subdivision of space. Particles are explicitly represented by their position, shape, and orientation.

Integration in OMNIS™

Mpacts is now integrated into the OMNIS™ environment: OMNIS™/Mpacts combines the power of the advanced DEM solver Mpacts with the multilingual user-friendly OMNIS™ interface and all its capabilities from the pre-processing to co/post-processing in one workflow. Furthermore, OMNIS™/Mpacts includes automatic Python scripting, multi-platform support, high-performance computing capabilities, and much more.

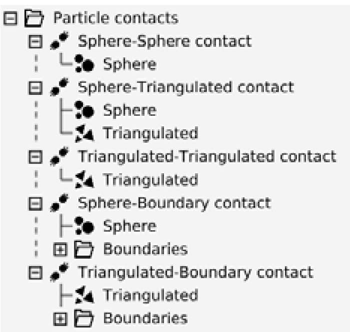
Thanks to OMNIS™/Mpacts, a DEM simulation has never been so easy to set up. From the setup of the mechanical motion of the geometry to the particle definition, all dedicated features are easily accessible and straightforward to use in the interface. To demonstrate its workflow, let's look in more detail at the ribbon mixer example.

Ribbon mixer setup in OMNIS™/Mpacts

In the Geometry context, any industrial geometry can be imported: STL, Parasolid, CATIA, etc... For the ribbon mixer we imported the geometry of the helicoidal rotor and the container. The mechanical motion of the rotor was set through a rotation Degree of Freedom (DoF). The rotation DoF was defined as a function of time to simulate the start-up of the ribbon rotation. Another option in OMNIS™ would have been to use the definition of translational motion.

In the Domain context, the domain of the simulation was defined (for rotor and stator). Several domains with modular geometry parts can be defined here, making it easy to simulate multiple designs of the ribbon mixer. And no mesh is needed, as the DEM method does not require any discretization. This is one of the main advantages of the MPacts solver, as it saves a lot in engineering and CPU time.

In the Simulation context, two particle classes were first defined: spheres and triangulated particles. Their properties were set through their material, motion integration (rotation and translation), and gravity parameters. Both the rotor and stator had their material defined as aluminium. Second, control volumes to generate more or less particles in the simulation domain were linked to the corresponding particle classes, in this case one for the spherical particles and another for the triangulated ones. They were both located on top of the mixer. The particle contacts were then automatically generated for each and between each particle class, resulting in five particle contacts for the ribbon mixer:



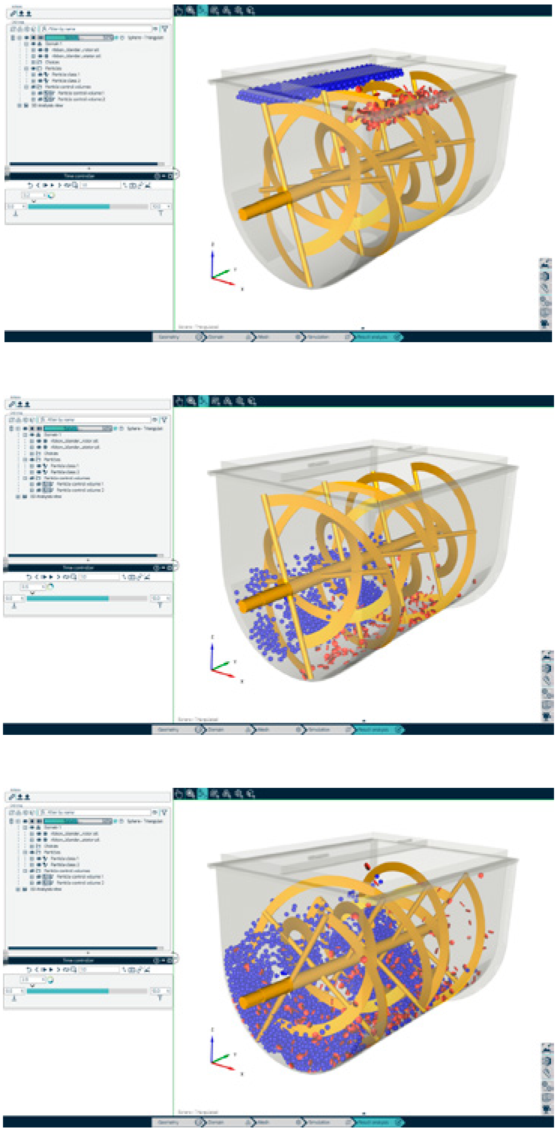
Each particle was assigned a contact law with its own properties.

OMNIS™/Mpacts also offers the possibility to define a particle injector that injects particles with a certain velocity into the simulation domain. But for the ribbon mixer application this functionality was not necessary.

In the Result analysis context, all dedicated post-processing functions are available for analyzing particle motions and their interactions. The evolution of the mixing process can be monitored in time to study the start-up of the rotor and the impact on the mixing of the particles.

Within the OMNIS™ workflow, different ribbon mixer designs and settings can be computed in a matter of minutes.

FIGURE 2 : Evolution in time of the ribbon mixer simulation.

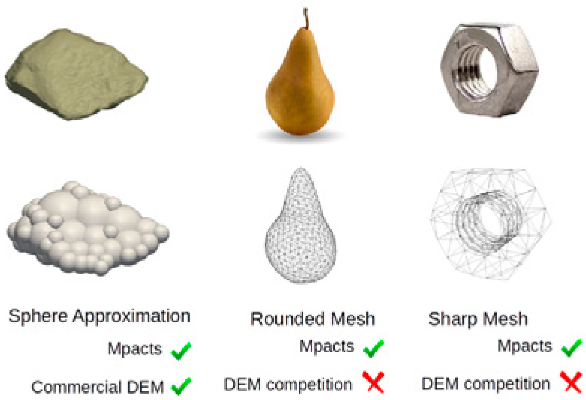


Not only spheres!

Primary DEM formulations assume particle shapes to be perfect spheres, as this method requires minimal computational effort. However, as illustrated in the ribbon mixer application, the ability to simulate multiple-shaped particles is essential to meeting industry requirements.

This is where Mpacts offers another unique and significant advantage: it can represent all possible particle shapes, preserving their physical properties entirely. Spheres, cylinders, polyhedra, capsules, fibers, sheets, and vesicles; they are all available in both rigid and deformable forms in Mpacts.

FIGURE 3 : OMNIS™/Mpacts can represent all possible particle shapes.



Sticky Walls / Particles
Void fraction

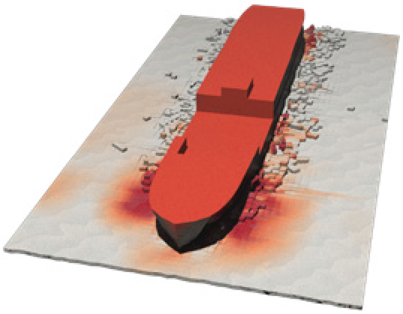
More applications

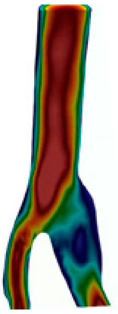
The Mpacts solver is integrated into OMNIS™ version 5.1. This version is focused on mechanical agitation applications such as mixers, conveyor belts, dump trucks, screw conveyors, etc. Its integration is continuous and some of the expert features described below will be released in future versions.

Without the need for building several experimental prototypes, industries can now simulate mixing processes in a couple of minutes for several designs in a user-friendly and powerful interface. OMNIS™/ Mpacts offers an all-in-one solution technology for any particle interaction application.

Damage modeling

How a product gets damaged or why it breaks apart is often important to know for robust and high-quality production. Mpacts therefore includes damage/breakage models that compute the onset and extent of damages. Custom damage/break models are also available for dedicated mechanisms.





Fluid velocity

CFD coupling

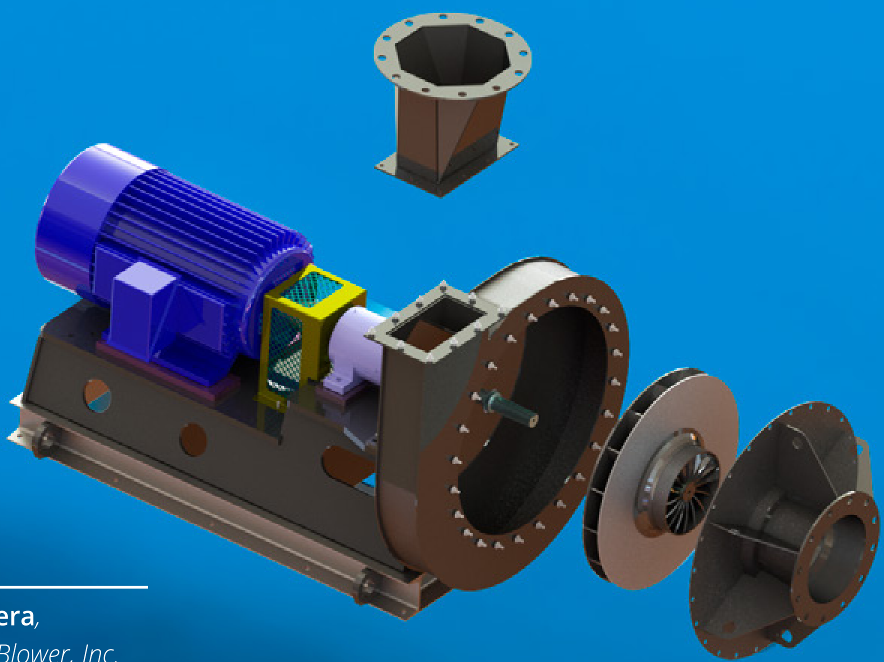
The study of fluid flows is essential for many industrial processes. A typical example is the simulation of the flow in an artery bifurcation for the biomedical industry. In this simulation, particle movement and collisions are performed using the DEM, while the fluid is simulated using CFD. Two-way coupling is performed. The particles influence the flow pattern: the flow will look for the 'easiest' path through the particle stacking, and through drag forces, the flow will also drive the motion of the particles.

Rigid-body coupling

Several parts of a machine are often mechanically linked with coupled motions. The coupling is usually done through rotational or translational constraints or a combination of both. The rigid-body functionality of Mpacts ensures a straightforward and smart definition of a coupled-motion machine, while maintaining high-speed simulations.

Increasing the performance of industrial blowers

with NUMECA optimization solutions



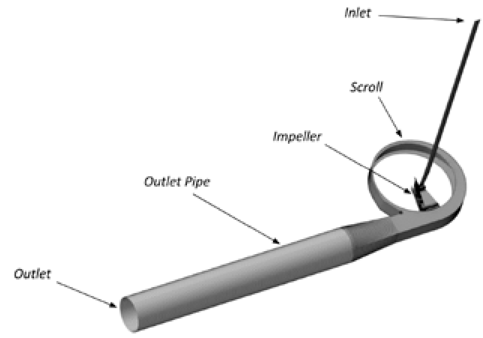
By **Dr Edward De Jesús Rivera**, *Engineering Manager, Illinois Blower, Inc.* and **Fanny Besem-Cordova**, *Application & Consulting Engineer, NUMECA-USA.*

Petrochemical and other industrial applications rely on fans for a series of gas handling processes - which can range anywhere from ambient air ventilation to harmful chloride gases above 500C. The maximum operation speed of these fans is governed and often limited by industry standards. This speed limitation forces fan manufacturers to size the diameter of the impeller up or down in order to obtain the desired pressure output. The challenge for engineers is that the resulting design can become oversized, heavy and expensive to fabricate, test and transport.

Illinois Blower, based in Cary, Illinois, has been working with NUMECA on a design approach that solves this dilemma by achieving higher pressures and increased fan efficiency, yet maintaining impeller speed and diameter. For more than 40 years, Illinois Blower has successfully developed and built custom centrifugal fans and blowers for a variety of worldwide industrial process industries, including refinery and petrochemical power generation, pollution control, pharmaceutical, food processing, and many others.

The goal of this particular case was to increase the pressure ratio of a complete fan stage (wheel and volute) over its entire performance line. Due to manufacturing constraints the solid body thickness around the impeller had to be maintained, and the blade shape needed to be easy to manufacture. In addition to this impeller optimization, the engineers wanted to get a better understanding of the flow physics to help reduce pressure losses in the outlet pipe.

FIGURE 1 : Test case: Impeller and volute of the centrifugal industrial fan to optimize with the surface mesh from Hexpress™.



"I have worked with NUMECA on several projects through the last eight years, and always appreciated the collaboration and the experience of the NUMECA-USA team, in particular in turbomachinery design optimization of torque converters and industrial fans. Such a project always involves objectives and constraints, which are quite intricate, and it is usually quite impossible to know from the project start how things will go. But what we have always done with the NUMECA team is work together, analyze situations, and decide on next steps. And I could count on them to find a way to deliver."

Edward De Jesús Rivera, Engineering Manager, Illinois Blowers.

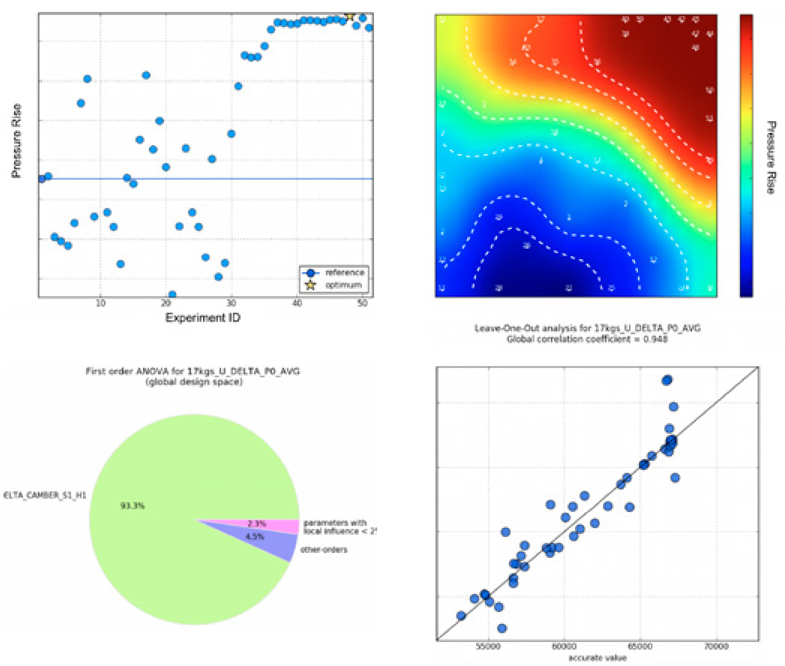
Methodology

A first series of simulations were run from choke to stall in FINE™/Open, NUMECA's unstructured multi-purpose CFD solver package, and the results were compared with experimental data to assure that the CFD settings were reliable.

High speed meshing

A 7.5M-point mesh with smooth boundary layers in the volute and around the impeller blade details was generated in Hexpress™. Only one passage of the impeller had to be meshed in order to run the performance line steady calculations, and each computation took less than 1 hour on 96 cores to achieve full convergence.

FIGURE 2 : From left to right, top to bottom: The scatter plot shows the value of an objective for all the database and optimization samples, with the optimum individual highlighted by a star. The self-organizing map reveals correlation and anti-correlation between parameters. The analysis of variance (ANOVA) decomposes the global variability of an output over a range of input variables. The Leave-One-Out (LOO) plot estimates the model reliability.



Optimization part 1: Identifying the main performance parameters

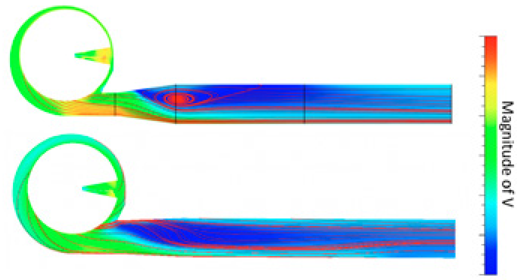
A first step of the optimization was to identify the main factors influencing the centrifugal fan's performance, in order to know exactly where optimization would be most effective. Twenty user-defined parameters of the impeller blade and flow channel were carefully selected, describing the hub and shroud shapes; the blade metal angles; and the blade camber and lean. The choice of these free parameters and associated variation ranges turned out to be key for the success of the project. For each set of the parameters a new geometry was created by FINE™/Design3D, and an unstructured mesh was automatically generated in OMNIS™/Hexpress, using a dedicated python script to save time.

After parameterization, the design of experiments (DOE) database was generated by FINE™/Design3D with the Minamo module. The main optimization algorithm of the Minamo data-mining tool is based on evolutionary algorithms, accelerated by the use of surrogate models to speed up the convergence rate. A database of 70 samples was built, filling the design space with 210 CFD solutions at three operating conditions (at stall, near design point, and at choke). By applying the grid-to-grid interpolation to improve the initialization of each CFD sample, a 25% reduction of iterations (and therefore CPU time) was achieved.

A thorough analysis of the database allowed the engineering team to understand the influence of each of the free parameters and their impact on the performance.

It was found that the volute was the main limiter in the optimization of the fan performance! Ensuing work de-coupled the components to further optimize the impeller separately and to design a new volute that minimizes the pressure losses observed in the sample CFD solutions.

FIGURE 3 : Comparison of the velocity streamlines and contours of velocity magnitude before and after volute redesign for one operating point.



Optimization part 2: Focus on the impeller

The second part of the performance optimization focused on the impeller blade, meridional effects, and corresponding solid walls, decoupled from the volute. Using Autogrid™ and FINE™/Turbo, the large database and optimization can easily be run over a weekend on a desktop machine, and the process is also fully automated by python scripts.

Once a new optimal impeller was obtained, the performance of the entire fan was then computed by coupling the detailed geometry of the optimal impeller with the redesigned volute.

FIGURE 4 : Static pressure difference for the baseline test data and the optimized CFD designs (values are removed for proprietary reasons).

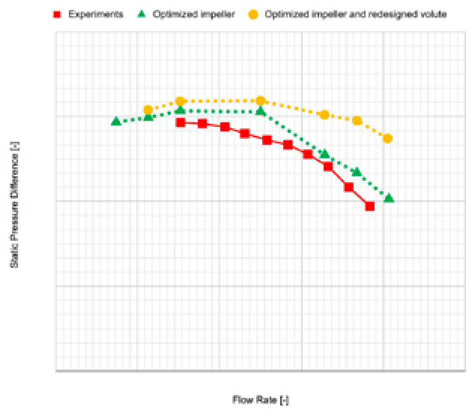
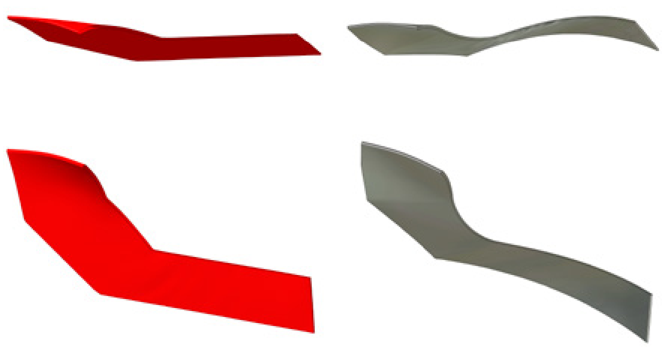


FIGURE 5 : Comparison of the impeller blade shape before optimization (in red) and after optimization (in grey).



Conclusion

Challenge completed. Thanks to this two-part optimization project, Illinois Blower managed to increase fan performance by up to 44% overall, while maintaining their original design constraints. The optimization of the impeller blade shape and flow channel led to an increase of static pressure of up to 20% at some operating points (near choke). Furthermore, thanks to a better understanding of the flow-induced pressure losses downstream of the volute scroll, a smart redesign of the volute led to an additional increase in pressure over the whole performance line, going up to 24% at some operating points (near choke).

> “A very successful alignment of the simulation data with the 17m sea trial data (within 1%) gave Teknicraft a high degree of confidence and understanding to run a design optimization study for the new catamaran.”
>
> *Whale Watch Kaikōura Ltd.*

A new era of sustainable ecotourism through design optimization of whale watching vessels

By **Nic de Waal**, *Managing Director, Teknicraft Design.*

Based in New Zealand on the Kaikōura Peninsula, Whale Watch provides sustainable ecotourism services for exploring and observing whales, dolphins, seals, and other mammals in their natural habitat. The tour company champions ecological preservation by using vessels specifically designed and constructed to operate in harmony with the mammals and their marine environment.

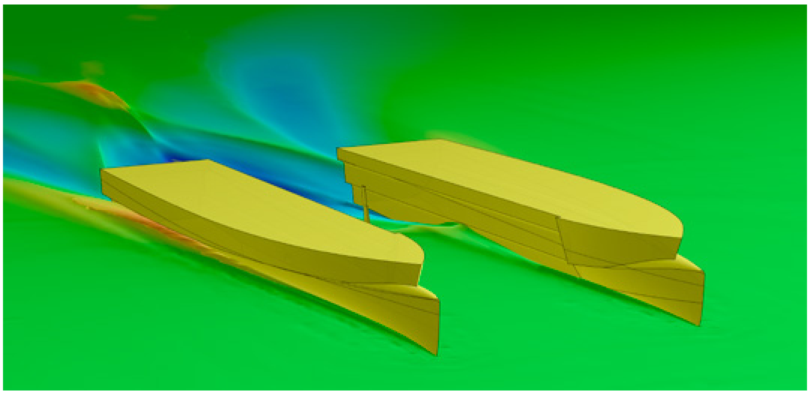
In their early years of operation, Whale Watch used a rigid hull inflatable, but in 1999 local naval architects Teknicraft designed their first catamaran to specifically address the requirements of excellent ride comfort and low fuel consumption combined with a superstructure arrangement suited for whale watching tours. Since the whales could be anything between 5 nautical miles and 30 nautical miles offshore on any given day, a vessel with high speed was required to spend the minimum time transiting and maximum time observing whales.

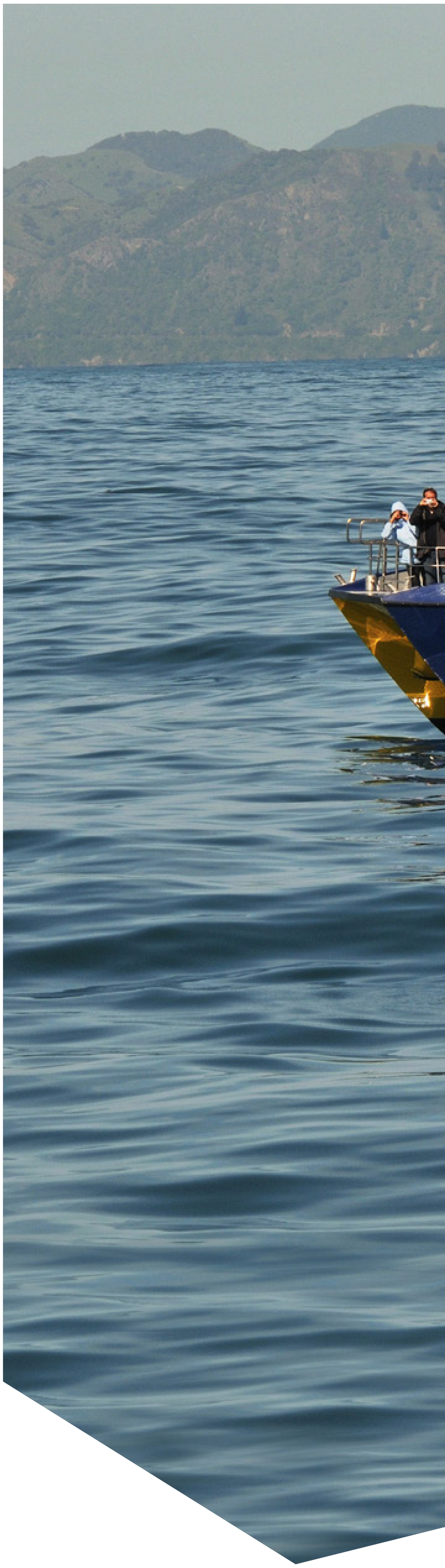
The Kaikōura Peninsula is directly exposed to the Pacific Ocean with no natural barriers, and back in the day the small, semi-protected natural harbor offered only limited space for vessel moorings during the day. This meant sightseeing craft had to be relatively small, challenging the requirements of a vessel that needed to operate in the seaways of the open ocean.

Teknicraft's solution for these challenges was a 17m-catamaran hull with hydrofoil support that provided lift to reduce pressure and friction resistance, while providing motion damping to increase the seakeeping and ride qualities.

Following the success of the first catamaran, a further five sister ships were built over the ensuing years. As the Whale Watch operation began to expand dramatically, larger vessels were discussed, but the depth and size of the natural harbor remained a limitation.

FIGURE 1 : Free surface elevation in the wake of the catamaran.





However, in November 2016, a massive earthquake hit the region, causing large parts of the shoreline to rise by approximately 1m, including the Kaikōura harbor. This overnight reduction in the water depth inside the harbor made it impossible for the fleet to operate. With government support, an extensive re-build of the Kaikōura harbor followed, making it larger and deeper and ensuring that it was now suitable for larger vessels. The deeper harbor presented new opportunities to Whale Watch, and Teknicraft was again commissioned, this time designing a larger, 24m-vessel that could operate at speeds between 30 and 35 knots, while carrying 120 passengers.

For their post-earthquake design Teknicraft decided to use Quad Hamilton HJ364 jet units because they are quieter for marine mammals compared with other forms of propulsion. Their propellers are also internal, so the risk of a propeller strike with marine mammals is completely eliminated.

They retained the hydrofoil support system, however the new foil was made adjustable in terms of its angle of attack. This control directly results in lower net fuel consumption as the foil can be set to its most effective position to suit different passenger and fuel loadings, as well as different operating speeds and sea conditions.

Teknicraft used FINE™/Marine for the optimization of the hull and the foil system. As the service speed of the new design operates at a high Froude number of 1.2, an initial calibration study was undertaken to align sea trial data from the 17m design with the parameters to be used in the simulations for the new vessel. Curve, surface and box refinements were used to ensure the required resolution was achieved to capture potentially complex turbulence formations in the region of the foil, chine and transom wakes, and a higher than normal mesh diffusion was used in the domain.

A very successful alignment of the simulation data with the 17m sea trial data within 1%, gave Teknicraft a high degree of confidence and understanding to run a design optimization study for the new catamaran.

Whale Watch's new 24m cat was launched in October 2020 and heralds in a new era of sustainable ecotourism for this innovative marine tourism operator. FINE™/Marine has played a significant part in the design's evolution, and the whole visitor experience will be heightened by this larger, more configurable and more comfortable vessel.

FIGURE 2 : Whale Watch's new 24m active foil assist catamaran.

Reducing Design Time with CloudPower

Supported closely by Numeca, Teknicraft, an early adopter of cloud computing, worked to establish optimal CloudPower settings out of the new South Pacific data center based in Sydney. CloudPower is a containerized virtual machine and allows on-demand access to a massively parallel HPC platform.

GPU-based containers were found to be ideal for meshing and results analysis, offering low latency and high GUI performance. For running simulations, CPU-based containers with a larger core density provided the floating point power to deliver results in the shortest possible time frame.

For the Whale Watch design optimization, Teknicraft ran a variety of load cases, trim conditions and foil angle of attacks. By opting to use FINE™/Marine in the cloud solver, time was minimized and iterations in the design loop ran over just hours and not days--all without the need to invest and maintain capital intensive multi-core workstations back at the office.

Particle counters and the COVID-19 pandemic

revolutionizing design through advanced simulation

The COVID-19 pandemic has brought into sharp focus the need to understand transmission mechanisms of respiratory viruses. Understanding transmission mechanisms requires study on three broad fronts: identification of virus transmission paths; establishing how the virus circulates; and, experimental validation of transmission and circulation models. Prior to the spread of COVID-19, in preparation for an anticipated influenza pandemic, the scientific community has shown that the short-range aerosol route is an important, though often neglected, virus transmission path.

Industry trade associations are collaborating to develop guidelines on how aerosol particles circulate through a building. To aid in their work, the trade associations are studying data provided by research groups who work in collaboration with particle counter manufacturers. Particle counters, typically used to measure the number of airborne particles in cleanrooms, research labs, and operating rooms, are now emerging as a technology that can help in determining behavior of aerosols.

By **Geoff Sheard**, *President, AGS Consulting, LLC.*

"Our objective in working with AGS and NUMECA was to predict particle motion within the particle counter itself, facilitating optimization of the particle counter geometry. Our experience is that better design minimizes recirculation of particles within the particle counter, irrespective of particle size."

Adam Giandomenico, President, Particles Plus.

A challenge that particle counter manufacturers are addressing is the characterization of the aerosol particles that carry the COVID-19 virus. When an infected individual sneezes, coughs or breathes, aerosol particles of water are expelled. The virus travels suspended in these particles. Since these are water particles, they evaporate over time and, therefore, particle size changes. Hence, particle counter manufacturers are challenged to measure the number of particles, especially as those particles change size.

Accurately measuring the number and size of aerosol particles carrying the COVID-19 virus is critical to the validation of any transmission or circulation model. In response, Particles Plus®, an engineering and manufacturing company located in Stoughton, Massachusetts, sought to apply computational fluid dynamics (CFD) and single-particle tracking simulations the analysis of the flow-field within a particle counter. The company selected AGS Consulting, LLC to partner with, because of their reputation for product design and optimization. They selected NUMECA to perform the CFD analysis, because of their reputation for simulation tools that accurately predict real-world product performance.

Working collaboratively with AGS Consulting, LLC and NUMECA, Particles Plus® commenced development of their next generation of particle counters.

Methodology

The numerical process adopted comprised two steps: first, modeling the particle counter flow-field that constitutes the particles' support media; second, calculating the trajectory of particles as they pass through the particle counter.

Due to the complexity of the geometry, engineers at NUMECA chose to work with FINE™/Open with OpenLabs, using the unstructured meshing capabilities of Hexpress™. First, for comparison with experimental results, flow-field simulations were run in FINE™/Open to predict pressure-drop through the particle counter. Here a pressure-based solver was used, which is faster and more accurate for incompressible flow-field simulations.

FIGURE 1 : Particle trajectories presented over two planes through the center of the particle counter: vertical Cut 1 and horizontal Cut 2.

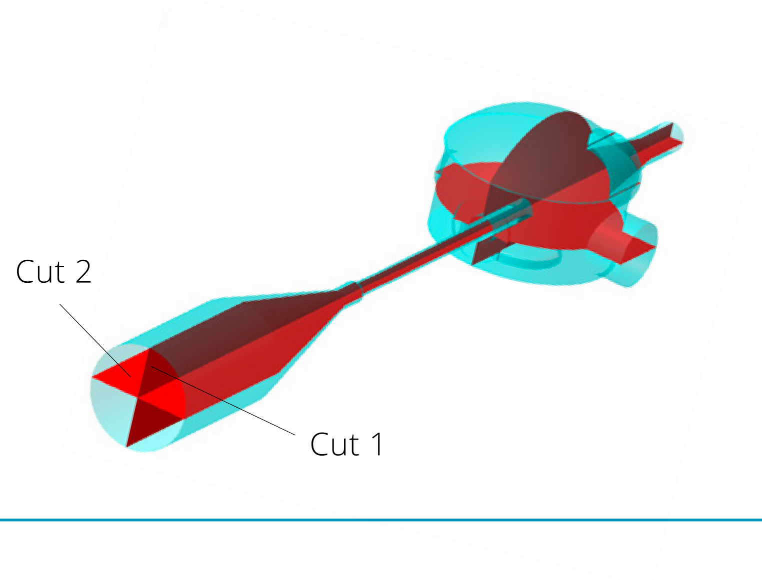
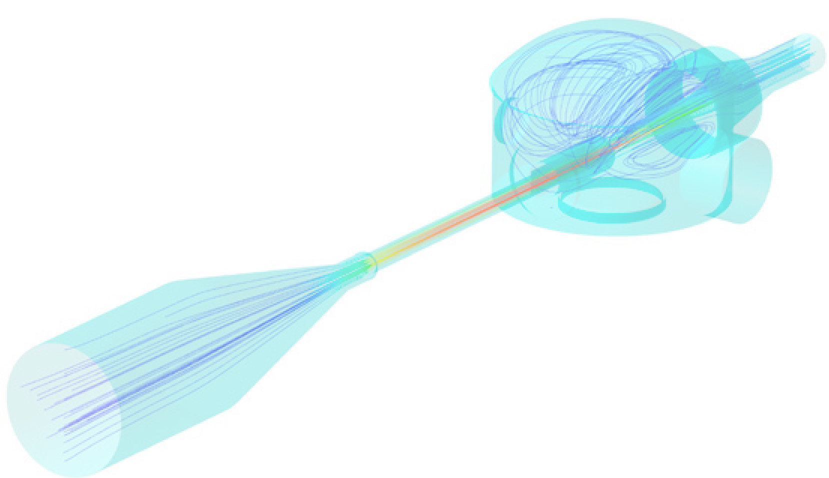


FIGURE 2 : 3D streamlines showing particle trajectories.



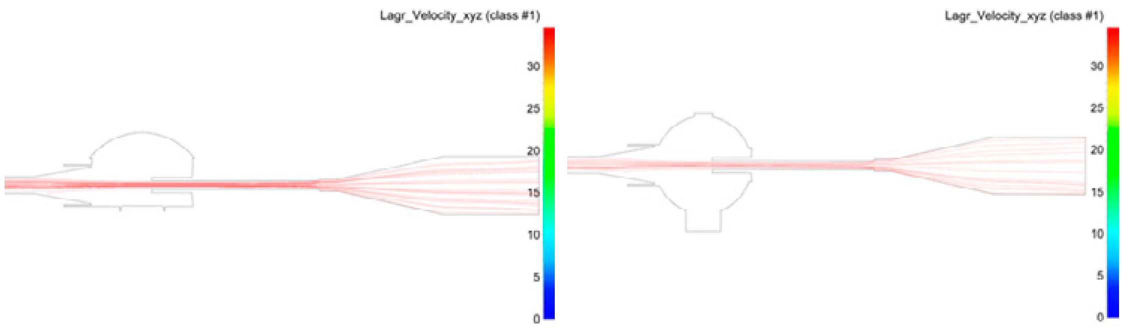
Single particle tracking was used to predict the trajectory of different sized particles through the particle counter. Particles were launched from the inlet, and trajectories calculated as they passed through the particle counter chamber.

Two assumptions were made when calculating trajectory:

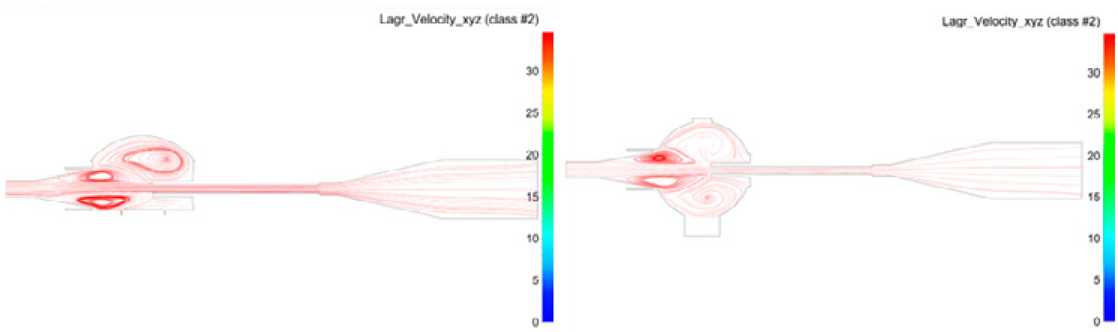
- » Particle trajectory is driven by the flow-field within the particle counter, but particles do not have a significant impact on the flow-field. The particle-to-air ratio is considered low enough for this one-way coupling approach to be assumed valid.
- » Particle-to-particle interaction is neglected. Again, the particle-to-air ratio is considered low enough for this approach to be assumed valid.

FIGURE 3 : Predicted trajectories for each class of particle over Cut 1 (left) and Cut 2 (right).

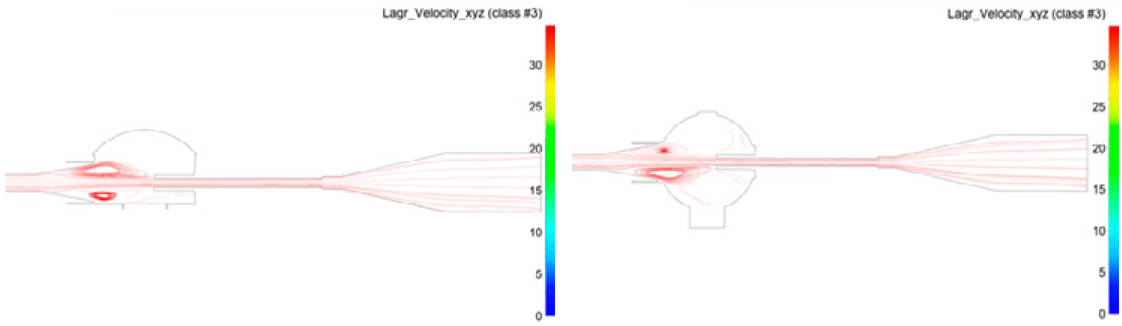
Class 1 (5.0 micrometer) diameter particles



Class 2 (1.0 micrometer) diameter particles



Class 3 (0.3 micrometer) diameter particles



Simulation Results

Horizontal and vertical cutting planes through the particle counter were defined and particle trajectories mapped onto each plane. The trajectories of Class 1, 2 and 3 particles were studied over the vertical plane, with each class of particle behaving differently:

» The Class 1 (5.0 micrometer) particles pass through the chamber.

» The Class 3 (0.3 micrometer) particles recirculated in the vicinity of the chamber exit. However, they do not migrate back into the main body of the chamber.

» The Class 2 (1.0 micrometer) particles both recirculated in the vicinity of the chamber exit and, critically, do migrate back into the main body of the chamber.

When particle trajectory was studied over the horizontal cutting plane, particle trajectories were concluded to be similar, with the exception of Class 2 particles. The Class 2 particles exhibited a more intense recirculation at the exit of the chamber. The intensity of this recirculation was tentatively concluded to be driving migration of Class 2 particles back into the main body of the chamber.

TABLE 1: Definition of Class 1, Class 2 and Class 3 particles (respectively 5.0, 1.0 and 0.3 micrometers diameter) used in the particle tracking simulation. As particle diameter reduces, the number of particles increases logarithmically.

Particle Class	Diameter	Volume	Volume Fraction	Concentration	Volume Concentration
1	5.0 E-06	7.0 E-17	1.00 E-04	5.0 E+00	3.0 E-16
2	1.0 E-06	5.0 E-19	9.05 E-07	6.0 E+00	3.0 E-18
3	0.3 E-06	1.4 E-20	2.66 E-08	6.5 E+00	9.2 E-20

Conclusions

Overall, AGS Consulting, LLC and NUMECA collaborated with Particles Plus® to develop an experimentally validated CFD simulation of the flow-field within a particle counter. This simulation was used to identify that large, medium and small particles behave differently as they pass through the particle counter. The two-step transfer of 1.0 micrometer particles from inlet-jet to exit-recirculation and then exit-recirculation to main-chamber recirculation, was not anticipated.

This insight into particle counter flow-field physics, and the associated physical mechanisms at play within the particle counter, have provided Particles Plus® with the basis for an on-going project aimed at identifying critical particle sizes prone to recirculation, and the optimization of particle counter geometry to minimize that recirculation.

AeroDelft pushes the airline industry towards a sustainable future with liquid hydrogen aircraft

By **Sam Rutten**, *Prototype Project Manager, AeroDelft*, **Olga Lubbers**, *Advisor, AeroDelft* and **Botond Pal**, *Application engineer, NUMECA International.*

AeroDelft is a student team at the forefront of sustainable aviation. While based in Delft, over forty students from different schools and universities around the Netherlands have joined together and are working hard to push the airline industry towards a sustainable future. AeroDelft has one main goal; to prove and promote liquid hydrogen as an alternative to conventional fuels in aviation. Air travel has been on the rise ever since it became a mainstream form of transportation, and it does not show any sign of slowing down.

> "To preserve the world we live in there are two options: stop flying or revolutionize the way in which we fly. AeroDelft aims to play a key role within that revolution."

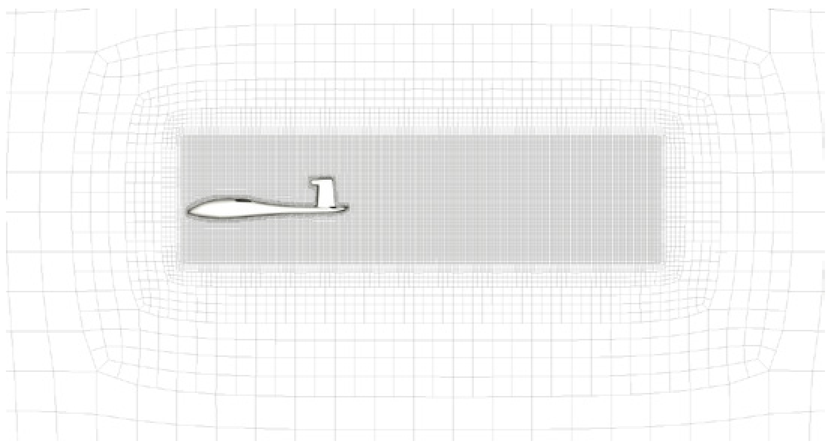
Currently, the team is working on two projects simultaneously. In 2018 Project Phoenix was founded. The team is now working on the final stretch of the Phoenix prototype, aiming to have the first successful liquid hydrogen powered flights in the summer of 2021. In addition, this year the Phoenix Full-Scale project has begun. This project will have a longer timeframe, but also has a major goal. By the year 2024 AeroDelft hopes to launch a full-scale liquid hydrogen powered airplane, which will be able to seat two people. From there on, the sky's the limit!

Getting the aerodynamics right for minimum energy consumption

One of the main requirements for developing hydrogen powered aircraft is to get the aerodynamics exactly right in order to minimize energy consumption. This is where AeroDelft's partnership with NUMECA comes in. To achieve the optimum aerodynamic design for their Phoenix prototype, AeroDelft uses OMNIS™ to be able to run many simulations and test a large variety of design alternatives until achieving the optimal result.

For the analysis described below, calculations for the aircraft were made based on cruise conditions with an inlet velocity of 23 m/s, cruise height of 100m, pressure of 101325 Pa and temperature of 15 degrees Celsius.

FIGURE 1 : Symmetry plane view with volumetric refinement visible.



Meshing and set-up

To meet the need for preliminary lift and drag estimates, OMNIS™ was used to run four simulations with increasing mesh resolution. Limited hardware use and simulation time were critical requirements.

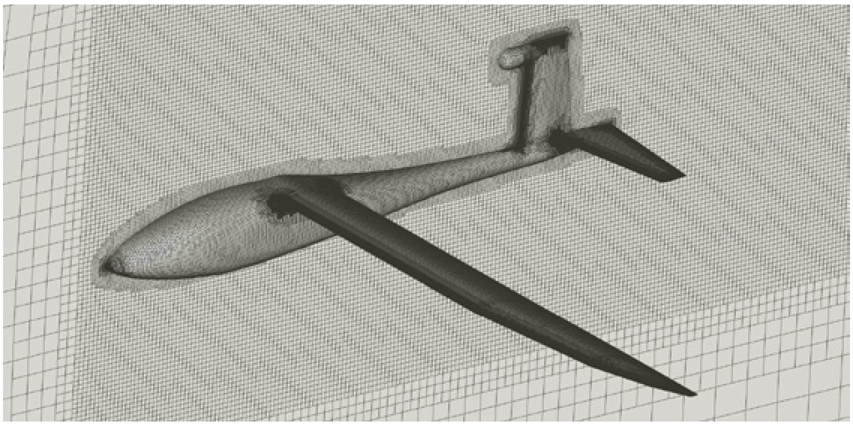
Starting from a CAD file, three fully hexahedral meshes were created with OMNIS™/HEXPRESS and a simulation was run on each of them. The first mesh was not refined within the boundary layer. In further mesh refinement, viscous layers were inserted to model the physical boundary layer with increased accuracy using a wall function. Additionally volume refinement was applied to investigate its influence on the solution. This is visible in Fig. 1.

The geometry of the aircraft CAD file included many small surfaces that were not very relevant for this analysis. Resolving these surfaces would normally drive up the cell count considerably, but with OMNIS™/HEXPRESS these regions can easily be meshed using a much larger cell size.

A good mesh quality was achieved with basic meshing settings and without any special treatment or refinement in geometrically complex areas. A closer view of the surface a of the aircraft and the mirror plane are visible in Fig. 2.

OMNIS™/HEXPRESS was able to compute an 8.5 million cell mesh in 22 minutes on 10 cores using an Intel® Core™ i9-7920X CPU. A simulation on the same mesh and hardware was completed in 2 hours and 13 minutes using CPU Booster technology. This is the equivalent of 2.6 core-hrs per million cells.

FIGURE 2 : Isometric view of aircraft surface mesh.



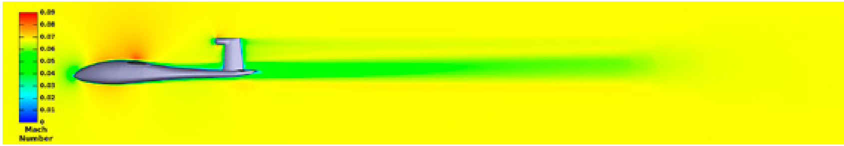
Simulation and Results

Simulations were run using the k-ωSST turbulence model. A 1:3 scale model of the aircraft was studied in cruise conditions. Half of the geometry was simulated using a spherical domain.

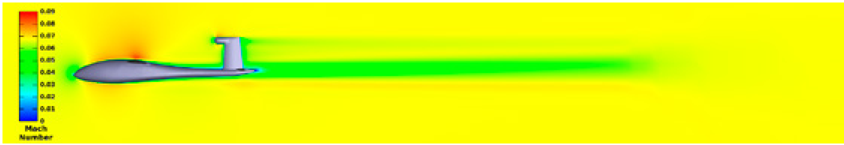
The first mesh dependency study provided a clear overview on the importance of mesh volume refinement for aerodynamics simulations. Fig. 3 shows the Mach number on the symmetry plane for the three meshes. The meshes with volume refinement clearly show better capturing of the aircraft wake than the coarse mesh without volume refinement.

FIGURE 3 : Mach number on the symmetry plane.

a) Finest mesh with volume refinement and 20 viscous layers



b) Fine mesh with volume refinement and 5 viscous layers



c) Coarse mesh without volume refinement and 5 viscous layers

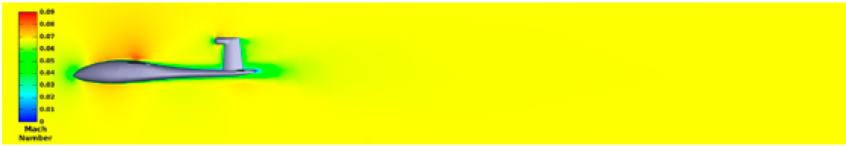
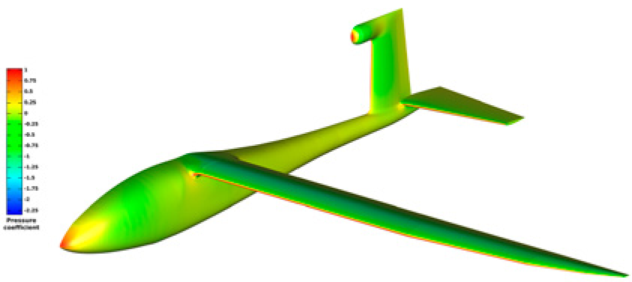


TABLE 1 : Lift and Drag results.

Mesh description	Finest	Fine	Coarse
# Viscous layers	20	5	5
Volume refinement in wake	Yes	Yes	No
Lift (% difference)	Reference	+2.4	+2.0
Drag (% difference)	Reference	-0.4	+2.7

Increasing the number of viscous layers had a significant impact on the predicted lift value. Such sensitivity studies are useful in identifying refinement areas that have the highest impact on the results for the lowest increase in computing cost. Table 1 shows an overview of the results.

FIGURE 4 : Pressure coefficient on the surface of the aircraft.



Furthermore, the pressure coefficient on the surface of the aircraft was calculated using the finest mesh as can be seen in Fig. 4. Non-dimensional quantities such as the pressure coefficient are very useful in scaling studies and comparisons with experimental results.

This project is an example of the fact that it is possible for student teams **to perform numerical studies with a significant impact at a reasonable computational cost**. Subsequent studies could explore the sensitivity of the results to further refinement at the leading/trailing edge of the wing and the vertical/horizontal stabilizer or the wake of the aircraft.

Lowering fuel consumption through Robust Design Optimization of ship propellers

By **Dr. Dirk Wunsch**,
*Head of Robust Design Group,
NUMECA International.*

Optimizing the design of ship propellers is of key importance for the marine industry, as it has a direct impact on operational costs through its influence on fuel consumption and ship performance. Even the smallest improvement in the design of the propellers can save ship operators millions of dollars across an entire fleet. The same is true, be it at a smaller scale, for the super yacht and pleasure boat industries for example.

Today propeller designs can be simulated meticulously with powerful tools that take into account a broad range of physics, from fluid flow to structural integrity to even acoustics behavior. And those designs can be optimized for optimal performance in a fully automated way.

The question that arises though, is whether we are sure that this 'optimal' design corresponds to the optimal design under real-world conditions, where uncertainties are an inevitable part of life. Uncertainties for example that are embedded in operating conditions such as ship speed and rotational speed of the shaft, or uncertainties in the manufacturing process, which lead to geometrical variations in the propeller shape that can have a significant impact on the final performance of the propellers.

To find a response to this question, robust optimization should be considered. Robust Design Optimization (RDO) takes into account a series of these uncertainties that can influence the performance of products. It allows for designs to be optimized in a 'robust' way, for example by making them less sensitive to inevitable variations in operating conditions or to small differences in geometries due to manufacturing variability.

To illustrate this, the below described customer case presents the optimization under uncertainties of a ducted ship propeller. The goal of this case was to reduce the impact of manufacturing variability on the ship's open water efficiency.

Methodology

Two optimization studies of a ducted ship propeller were performed. First, a standard (deterministic) design optimization was used to maximize the open water efficiency of the propeller. Then, in a second step, robust design optimization was performed to maximize the **mean value** of the open water efficiency and minimize standard deviation of this efficiency.

Uncertainty Quantification

A total of 12 uncertainties were identified and characterized for this case: 11 manufacturing uncertainties and 1 operational uncertainty (axial velocity). The manufacturing uncertainties were deduced from the technical norm ISO-484-2, which specifies all the manufacturing tolerances for this particular case. The manufacturing variability of every propeller has to respect these imposed manufacturing tolerances.

NUMECA's optimization software FINE™/Design3D is equipped with a unique Uncertainty Quantification (UQ) analysis module. It enables designers to easily assess the effects of uncertainties on performance by carrying out several CFD simulations in a fully automated way. In this study the 12 defined uncertainties resulted in 35 individual CFD simulations.

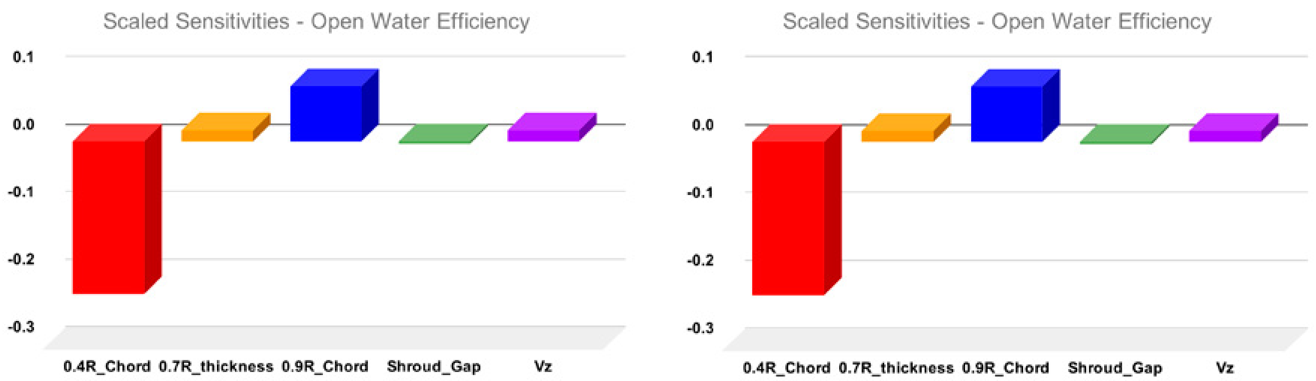
Scaled sensitivities

To gain physical insight into the influence of uncertainties on performance and to reduce computational cost, NUMECA developed an intelligent post-processing tool, called "scaled sensitivities". This tool measures the sensitivity of a performance, in this case open water efficiency, to specific uncertainties. Exploiting these sensitivity derivatives enables designers to reduce the number of uncertainties to be analyzed to the minimum relevant ones, and thus minimize computational cost.

For this study Figure 1 shows that open water efficiency is very sensitive to propeller chord length and ship velocity, while thickness plays only a minor role. Knowing this allows us to merge all thickness sections together for this study, forming one single uncertainty control for the thickness at 70% of the span height. This enables us to reduce the total number of uncertainties to be taken into account from 12 to 5, even though the total impact of all uncertainties is maintained in the analysis.

FIGURE 1 : Scale sensitivity on the open water efficiency.

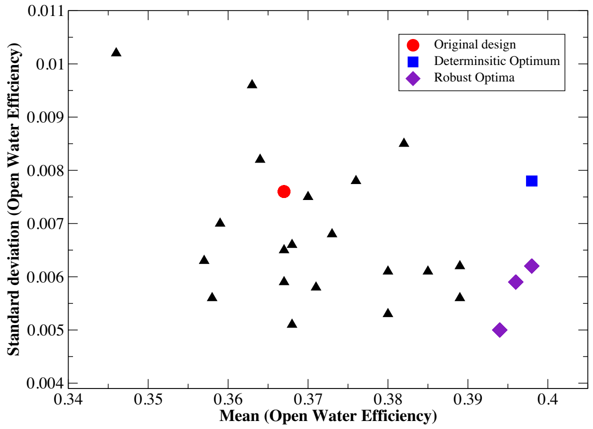
a) Full set of uncertainties | b) Equivalent reduced set of uncertainties



Deterministic vs robust optimal design

UQ analysis was performed on the standard deterministic design, in order to be able to compare its results with the robust optimal version. Figure 2 shows a characteristic Pareto plot where the standard deviation of the open water efficiency is shown over its mean value.

FIGURE 2 : Pareto plot showing the standard and robust optimal design together with the baseline design in the two objective spaces.



» Standard design optimization

The baseline design that the study started from is indicated in the graph by the red dot and the UQ results of the standard optimal design (not taking into account uncertainties) by the blue square. The plot shows that the open water efficiency has increased quite significantly by 8.5%, but that there is also an increase of 2.6% in its standard deviation. This means that this design is slightly more sensitive to the influence of manufacturing variability and axial velocity than the original design.

» Robust design optimization

Robust optimal design 3 plotted in fig. 2 shows that the mean value of the open water efficiency increased by the same amount as for the standard optimal design, namely 8.5%. However, its variability is reduced by -17.7%. That means that this design is less sensitive to the influence of manufacturing variability and axial velocity than the original design and that it provides more stable performances.

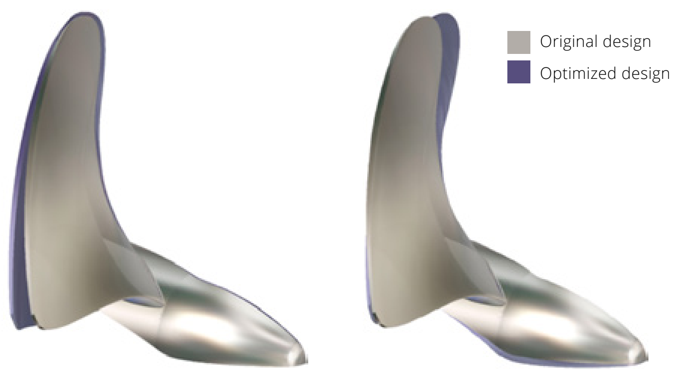
Propeller shape

Figure 3 compares the shapes of the standard optimized design and the best robust optimized result with the original propeller shape. Even though performance is the same for both designs, their shapes are significantly different!

FIGURE 3 : Resulting propeller shape.

a) Standard deterministic design

b) Robust design



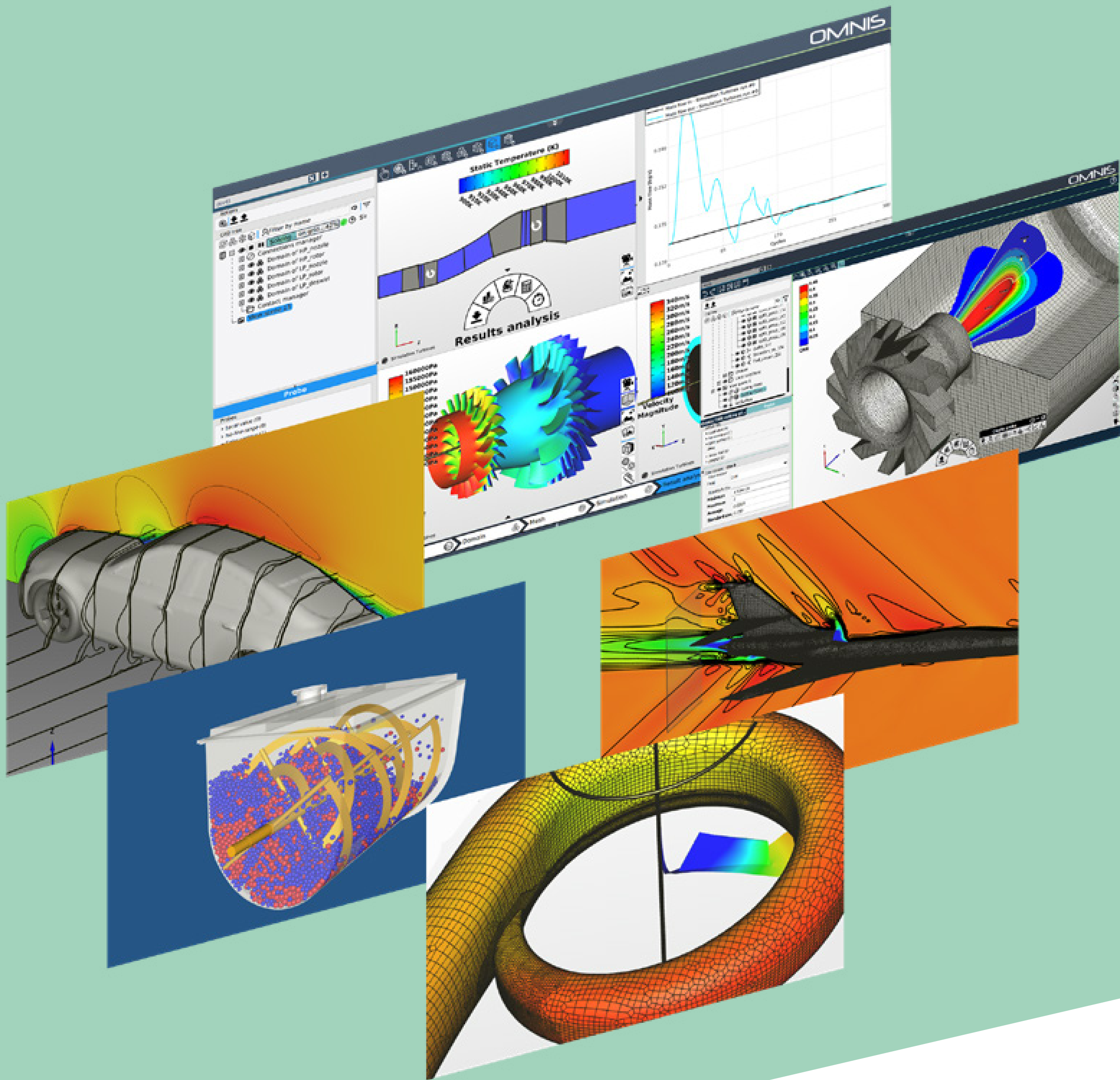
Conclusions

Robust Design Optimization enables engineers to create designs that are less sensitive to existing and unavoidable manufacturing and operational variabilities. Comparing standard and robust design optimization clearly showed that a comparable performance increase can be achieved with both strategies in this marine propeller case, but only the robust optimization allows for reduction of performance variability, making it less sensitive to uncertainties originating from the manufacturing process or from operational variability.

OMNIS

Productivity - Reliability - Usability

www.numeca.com/products/omnis



NUMECA International

Ch. de la Hulpe 189 Terhulpse Steenweg
1170 Brussels - Belgium
+32 (0)2 647 83 11

www.numeca.com
Contact us: info@numeca.be

NUMECA INTERNATIONAL

Ensuring your competitive advantage